

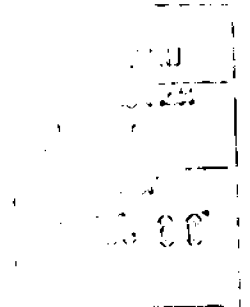
MODELING AND SIMULATION OF AN ASYMMETRIC PLANE DIFFUSER

SITI QHADIJAH BT ABDUL RASHID

**A report submitted in partial fulfilment of the
requirements for the award of the degree of
Bachelor of Mechanical Engineering**

**Faculty of Mechanical Engineering
Universiti Malaysia Pahang**

NOVEMBER 2007



ABSTRACT

This thesis deals with modeling and simulation of an asymmetric plane diffuser which is the most interesting test case in flow field by using Fluent software. The objective of this thesis are to understand the physical of the flow field in the plane diffuser, to evaluate turbulence model performance in predicting flow field using CFD commercial code, FLUENT and to compare with experimental data. The flow behavior includes pressure distribution and velocity profiles in asymmetric plane diffuser was investigated using three turbulence modeling which are realizable k- ϵ model, standard k- ω model and SST k- ω model. The results from the simulation was compared to the experimental data (Buice & Eaton 1997) in order to produce accurate result. The acquired results of pressure distribution shows that both k- ω model provide a fairly accurate in beginning stage while realizable k- ϵ shows the worst result. For velocity profiles within the diffuser, the simulation results are much different from the experiment except the bottom half of the diffuser. The realizable k- ϵ model being the worst followed by standard k- ω model while SST k- ω model gives better result. For the recirculation region, SST k- ω model predicts the biggest region while realizable k- ϵ model shows the smallest. The CFD commercial codes can solve problems faster and the best way to determined the fluid flow in this project.

ABSTRAK

Tesis ini memperkatakan tentang pemodelan dan simulasi pada penyebar kapal tidak simetri dimana ia adalah kes ujian yang paling menarik dalam bidang aliran dengan menggunakan perisian Fluent. Objektif tesis ini adalah untuk memahami aliran fizikal di dalam penyebar, menilai pelaksanaan model kegeloraan dalam meramalkan aliran menggunakan perisian Fluent dan membandingkan dengan data eksperimen. Aliran yang bertindak termasuklah penyebaran tekanan dan profil halaju dalam penyebar diuji dengan menggunakan tiga model turbulent iaitu model $k-\epsilon$, model standard $k-\omega$ dan model SST $k-\omega$. Keputusan dari simulasi di bandingkan dengan data eksperimen (Buice & Eaton 1997) untuk menghasilkan keputusan yang tepat. Keputusan yang diperolehi daripada penyebaran tekanan menunjukkan kedua-dua model $k-\omega$ menghasilkan keputusan yang agak tepat pada peringkat awal manakala model $k-\epsilon$ menunjukkan keputusan yang tidak memuaskan. Untuk profil halaju pula, keputusan simulasi sedikit berlainan daripada data eksperimen kecuali pada dasar penyebar. Model $k-\epsilon$ menunjukkan keputusan terburuk diikuti dengan model $k-\omega$ manakala model SST $k-\omega$ menunjukkan keputusan yang terbaik. Untuk kawasan pengaliran balik, model SST $k-\omega$ menunjukkan kawasan yang terbesar manakala model $k-\omega$ menunjukkan kawasan pegaliran balik yang terkecil. Perisian aliran (CFD) dapat menyelesaikan masalah aliran dengan cepat dan adalah cara yang terbaik untuk menentukan bidang aliran cecair dalam projek ini.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	TITLE PAGE	
	SUPERVISOR'S DECLARATION	i
	DECLARATION	ii
	ACKNOWLEDGEMENT	iv
	ABSTRACT	v
	ABSTRAK	vi
	TABLE OF CONTENTS	vii
	LIST OF FIGURES	x
	LIST OF ABBREVIATIONS	xi
	LIST OF SYMBOLS	xii
	LIST OF APPENDICES	xiv
1	INTRODUCTION	
	1.1 Project Background	1
	1.2 Problem Statement	1
	1.3 Objectives of Project	2
	1.4 Scopes of Project	3
	1.5 Flow Chart	3
2	LITERATURE REVIEW	
	2.1 Introduction	4
	2.2 Computational Fluid Dynamics (CFD)	4
	2.3 Fluent	5

2.4	Gambit	7
2.5	Turbulence Modeling	7
2.5.1	The k- ε model	10
2.5.2	The k- ω model	11
2.5.3	Near Wall Treatment Condition	12
2.5.4	Discretization Method	13
2.5.5	Navier-Stokes Equation	16
2.5.6	Conservation of Mass	17
2.5.7	Conservation of Energy	18
2.5.8	Conservation of Momentum	19
2.5.9	Conservation of Angular Momentum	19
2.6	The Asymmetric Plane Diffuser	19
2.7	Test Case	21
2.8	Turbulence	22
2.9	Previous Study	24

3

METHODOLOGY

3.1	Introduction	26
3.2	Methodology Flow Chart	26
3.3	Define Problem	28
3.4	Gather Information in Literature Review	28
3.5	Design Process	28
3.6	Test for Grid Independence	29
3.7	Simulation Processes	30
3.7.1	Pre-processing	30
3.7.2	Processing/Solver	31
3.7.3	Post processing	32
3.8	Analyze	33
3.9	Verification and documentation	34
3.10	Conclusion	34

4	RESULTS AND DISCUSSION	
4.1	Introduction	35
4.2	Pressure Distribution	36
4.3	Velocity Profile	38
4.4	Position of Recirculation Region	44
5	CONCLUSION & RECOMMENDATIONS	
5.1	Conclusion	48
5.2	Recommendations	49
	REFERENCES	50
	APPENDICES A-E	51

LIST OF FIGURES

FIGURE NO	TITLE	PAGE
1.1	Project Flow Chart	3
2.1	The geometry of the asymmetric Plane Diffuser	22
2.2	Tracer transport in laminar and turbulent flow	23
3.1	Project Flow Chart	27
3.2	Simple 2-D mesh	29
4.1	Residual convergence	35
4.2	Countours Profile of Total Distribution	36
4.3	Graph Pressure Coefficient vs x/H	37
4.4	Velocity profile at $x/H = 03$ and $x/H = 06$	39
4.5	Velocity profile at $x/H = 13$ and $x/H = 17$	39
4.6	Velocity profile at $x/H = 20$ and $x/H = 24$	40
4.7	Velocity profile at $x/H = 27$ and $x/H = 30$	40
4.8	Velocity profile at $x/H = 34$ and $x/H = 40$	41
4.9	Velocity profile at $x/H = 47$ and $x/H = 53$	41
4.10	Velocity profile at $x/H = 60$ and $x/H = 67$	42
4.11	Velocity profile at the outlet	42
4.12	Graph Skin Friction Coefficient vs x/H	45
4.13	Recirculation region in the diffuser (SST $k-\omega$ model)	45
4.14	Recirculation region in the diffuser (standard $k-\omega$ model)	46
4.15	Recirculation region in the diffuser (realizable $k-\epsilon$ model)	46

LIST OF ABBREVIATIONS

CFD	-	Computational Fluid Dynamics
DNS	-	Direct Numerical Solution
LES	-	Large Eddy Simulation
RANS	-	Reynolds Average Navier-Stokes Equation
RSM	-	Reynolds Stress Model
SST k- ω	-	Shear Stress Transport k- ω Model

LIST OF SYMBOLS

P	-	Static Pressure
U, u	-	Fluid Velocity
U_b	-	Fluid Bulk Velocity
u	-	Mean Fluid Velocity Component
$u(t)$	-	Fluctuating Fluid Velocity Component
3-D	-	Three Dimensional
2-D	-	Two Dimensional
H	-	Height of Inlet of Diffuser
Re	-	Reynolds Number
ρ	-	Fluid Density
τ_w	-	Wall Shear Stress
μ	-	Kinematic Viscosity
μ_t	-	Turbulence Viscosity
k	-	Turbulent Kinetic Energy
ε	-	Dissipation of Turbulent Kinetic Energy
P_k	-	Production of Turbulent Kinetic Energy
C_μ	-	Turbulent Viscosity Constant
$C_{\varepsilon 1}$	-	Standard k- ε constant
$C_{\varepsilon 2}$	-	Standard k- ε constant
σ_ε	-	Standard k- ε constant
ϖ	-	Inverse Time Scale
β	-	Standard k- ϖ constant
σ_k^*	-	Standard k- ϖ constant
α	-	Standard k- ϖ constant
β	-	Standard k- ϖ constant

σ_w^*	-	Standard k - ω constant
y	-	Distance normal to the wall
C_f	-	Skin Friction Coefficient

LIST OF APPENDICES

APPENDIX	TITLE	PAGE
A	Gantt Chart for FYP 1 and FYP 2	51
B	The Basic Guide in Using Fluent	52
C	Defining Boundary Condition	54
D	Defining Node Value	55
E	Convergence Monitor : Residuals	56

CHAPTER 1

INTRODUCTION

1.1 Project Background

Flow in the asymmetric plane diffuser is considered by researchers as one of the most interesting test cases due to the flow characteristics that occur in the flow. These characteristics include flow separation of a fully developed turbulent flow, due to an adverse pressure gradient generated by the channel expansion. Flow reattachment, and redevelopment of the flow also occurs downstream of the expansion. The problem is taken as a general case of situations where separation occurs either on a flat plane or a gently curved surface. The problem is similar in nature to what happens on the suction side of an airfoil as it reaches stall conditions (separation), yet the geometry is confined and thus requires fewer nodes than the stalling airfoil problem. The results from this test case can thus be use to validate and verify turbulence models that can then be applied to these problems with confidence in the mechanical engineering sector. The project is entirely based upon the use of FLUENT.

1.2 Problem Statement

The experimental test case presented in this project is a separated flow in an asymmetric plane diffuser. A diffuser is the mechanical device that is designed to

control the characteristics of a fluid at the entrance to a thermodynamic open system. Diffusers are used to slow the fluid's velocity and to enhance its mixing into the surrounding fluid. In this plane diffuser, the critical part that to be analyze is at the inclined wall where the circulation of the fluid flow occurs. The degree of the tangential and the inclined wall is only about 10° . So, it is very difficult to determine the velocity and pressure of the fluid flow inside the plane diffuser by experimental way. It takes so much time and need to build the real plane diffuser. It is also very highly cost because if the error occurs, we need to do it all over again. Nowadays, the high technology of Computational Fluid Dynamics (CFD) can easily be used in order to simulate the fluid flow of the plane diffuser. It gives more advantages such as, not wasting so much time and low cost in doing this research. For example, if the errors occurs, we can easily change the method or some value until we get the satisfied value. Besides, we solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry. In this high technology world, people usually search for the effective, faster and accurate method. CFD commercial codes are the best way to determined the fluid flow in this project.

1.3 Objectives of Project

To measure the extent of one project should go; some objectives need to set in order to ensure the success of the project. Therefore, the objectives of this project are :

- i. To understand the physic of the flow field in the plane diffuser.
- ii. To evaluate turbulence model performance in predicting flow field using CFD commercial code, FLUENT.
- iii. To compare with available experimental data.

1.4 Scopes of Project

Scopes are important steps or procedures to be applied in achieving objectives. Based on objectives above, the scope of this project are defines as follow:

- i. Simulation studies using CFD commercial code in asymmetric plane diffuser.
- ii. Use the $k-\epsilon$, $k-\omega$ and SST $k-\omega$ turbulence model in FLUENT
- iii. Geometry setup and mesh models for test case using GAMBIT

1.5 Flow Chart

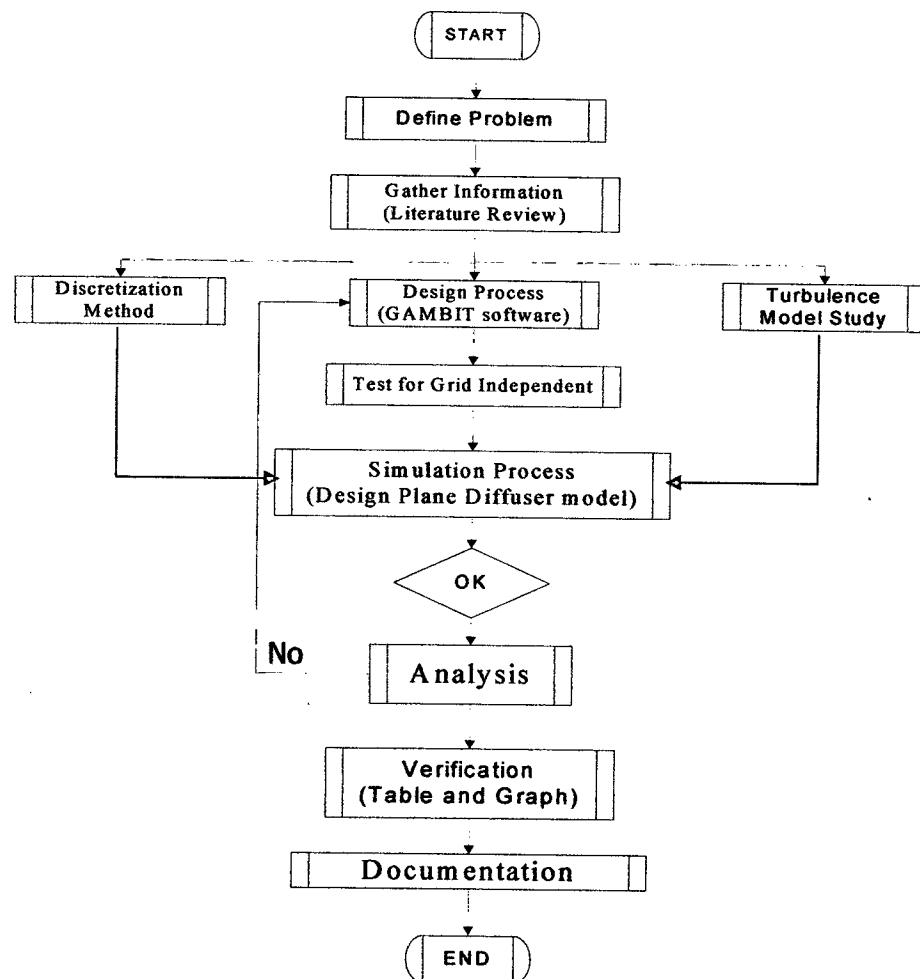


Figure 1.1 Project Flow Chart

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In order to perform this project, literature review has been made from various sources likewise journal, books and other references. The reference sources emphasize on few important aspects which are related to the asymmetric plane diffuser.

This chapter will described about asymmetric plane diffuser, turbulence, the near wall consideration in turbulence model, Computational Fluid Dynamics (CFD), turbulence modeling, discretization method and will be discussed about the test case of the plane diffuser.

2.2 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many

cases. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic or turbulent flows are an ongoing area of research. This technique is very powerful and spans a wide range of industrial and non-industrial applications areas. For examples in aerodynamics of aircraft and vehicles, hydrodynamics of ships, power plant, turbomachinery and many more. This software actually economical and sufficiently complete. There are several unique advantages of CFD over experiment-based approaches to fluid design :

- Substantial reduction of lead times and costs of new designs
- Ability to study systems where controlled experiments are difficult or impossible to perform.
- Ability to study systems under hazardous conditions at and beyond their normal performance limits.
- Practically unlimited level of detail of results.

The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any single-phase fluid flow.

2.3 Fluent

FLUENT is a renowned computer programme of modeling and simulating fluid flows and heat transfer problems in order to solve the fluid equations and produce the required solutions. FLUENT's pre-processor is another program called GAMBIT, which provides the tool of building and applying the mesh on the geometrical representation of the problem concerned. Since meshing is an important process of the CFD analysis, as too coarse a grid since would have very high errors in the solution and too fine a grid would only waste computational time, GAMBIT provides the user with easy commands of modifying the mesh as required. The processing and post-processing stages are done by FLUENT. This program enables the user to choose and set the discretization method, model material and

convergence criteria, among others, used for the analysis. Among commercial CFD software products, FLUENT has the largest array of industrially tested capabilities - some 1,000 physical models. These models are remarkably robust, with associated features to accelerate convergence every time. Because FLUENT uses unstructured, hybrid modeling technology, models can be built that conform to arbitrary geometric shapes and other complex surfaces. As a result, your CFD model will have accuracy it needs, where it is needed. These are the advantages of using FLUENT software:

i) Ease-of-Use

- FLUENT allows you to make changes to the analysis at any time during the setup, solution, or post processing phase. This saves time and enables you to refine your designs efficiently. The intuitive interface makes learning easy. Smart panels show only the modeling options that are appropriate for the problem setup at hand. CAD geometries are easily imported and adapted for CFD solutions.

ii) Speed

- Solver enhancements and numerical algorithms that decrease the time to solution are included in every new release of FLUENT software. Our mature, robust, and flexible parallel processing capability enables you to solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry

iii) Powerful Visualization

- FLUENT's post processing provides several levels of reporting, so you can satisfy the needs and interests of all audiences. Quantitative data analysis can be as rigorous as you require. High resolution images and animations allow you to communicate your results with impact. Numerous data export

options are available for integration with structural analysis and other CAE software programs.

2.4 Gambit

GAMBIT is Fluent's geometry and mesh generation software. GAMBIT's single interface for geometry creation and meshing brings together most of Fluent's preprocessing technologies in one environment. Advanced tools for journaling let you edit and conveniently replay model building sessions for parametric studies. GAMBIT's combination of CAD interoperability, geometry cleanup, decomposition and meshing tools results in one of the easiest, fastest, and most straightforward preprocessing paths from CAD to quality CFD meshes.

As a state-of-the-art preprocessor for engineering analysis, GAMBIT has several geometry and meshing tools in a powerful, flexible, tightly-integrated, and easy-to use interface. GAMBIT can dramatically reduce preprocessing times for many applications. Most models can be built directly within GAMBIT's solid geometry modeler, or imported from any major CAD/CAE system. Using a virtual geometry overlay and advanced cleanup tools, imported geometries are quickly converted into suitable flow domains. A comprehensive set of highly automated and size function driven meshing tools ensures that the best mesh can be generated, whether structured, multiblock, unstructured, or hybrid. GAMBIT's range of CAD readers allow you to bring in any geometry, error free, into its meshing environment. GAMBIT also has an excellent boundary layer mesher for growing optimum grid cells off wall surfaces in the geometries for fluid flow simulation purposes.

2.5 Turbulence Modelling

A turbulence model is a computational procedure to close the system of mean flow equations so that a more or less wide variety of the problems can be calculated. For most engineering purposes, it is unnecessary to resolve the details of turbulence fluctuations. Only the effects of the turbulence the mean flow are usually sought. For a turbulence model to be useful in a general purpose CFD code it must have wide applicability, be accurate, simple and economical to run. The most common turbulence models are classified as below:

1. Spalart-Almaras Model
2. Types of the k - ϵ model (standard, RNG, realizable)
3. Types of the k - ω . model (standard, SST)
4. Reynolds Stress Models (RSM)
5. Large Eddy Simulation (LES)

Fluid flows in practice generally can be divided into three types, i.e. creep, laminar and turbulent flow, with turbulent flow being the most common, complex and unpredictable of the three. This is due to the characteristics of these types of flows, which are listed below:

1. The inertia of fluid particles are more dominant than the viscous effects of the
2. Flow, causing it to be unstable
3. Instantaneous (“snapshots”) of turbulent flows are very unpredictable and ‘chaotic’ with a fluctuating velocity field, but there is hope to predict the mean
4. Values.
5. Almost all turbulent flows are three dimensional (3-D) flow as the fluctuations occurs rapidly in all three spatial dimensions

6. Occurrence of turbulent diffusion, which is the process of parcels of fluid with differing concentration of one or more of the fluid's transported quantities, mixes together as they are brought into contact
7. Mixing causes loss of mean potential or kinetic energy due to the action of viscosity, which is an irreversible process known as dissipation
8. The flows possess coherent structures, consist of repeatable and deterministic, often large scale events. However, these events occur randomly and differ each time, thus making it very unpredictable.

Thus numerical methods of solving these flows are essential and play a very important role in the design process. There are many approaches to solving turbulent flows, which include the use of correlations, the use of integral equations, Direct Numerical Solutions (DNS) and Large-Eddy-Simulation (LES), where the latter two attempt to produce a collection of “snapshots” of instantaneous flows, which can then be averaged similarly to experimental data. The one method that will be discussed at length is the Reynolds-Average Navier-Stokes (RANS) equation, which directly attempts to solve model equations for the mean quantities since this is the approach taken for this project.

Turbulent flows are characterized by fluctuating velocity fields. This causes the transported quantities such as momentum, energy, and species concentration to mix, and cause these quantities to fluctuate as well. Since the fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the fundamental equations can be time-averaged, manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities. This is demonstrated below. The instantaneous velocity, U , consists of a mean velocity, \bar{U} , and a fluctuating velocity, $u(t)$, given by

$$U = \bar{U} + u(t) \quad (2.1)$$

The Reynolds-averaged Navier-Stokes (RANS) equations are time-averaged. Equations of motion for fluid flow. They are primarily used while dealing with turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate averaged solutions to the Navier-Stokes equations.

For incompressible flow of Newtonian fluid, these equations can be written as,

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \frac{\partial \bar{u}_j \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[-\bar{p} \delta_{ij} + \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u_i u_j} \right] \quad (2.2)$$

The left hand side of this equation represents the change in mean momentum of fluid element due to the unsteadiness in the mean flow and the convection by the mean flow. This change is balanced by the mean body force, the isotropic stress due to the mean pressure field, the viscous stresses, and apparent stress $\rho \overline{u_i u_j}$ due to the fluctuating velocity field, generally referred to as Reynolds stresses. The presence of these terms causes the equation to be unsolvable since there are too many unknowns. Thus it is required to model these stresses, and this is where the turbulence models come in. The Reynolds stresses are modeled using a model termed the eddy-viscosity model, given by:

$$-\rho \overline{u_i u_j} = \mu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} k \quad (2.3)$$

μ_t is known as the turbulence viscosity, which can be characterized by two parameters, the turbulent kinetic energy, k , and a length scale, L . Thus, by modeling an equation for the kinetic energy, k , and the length scale, L , can be solved, and having done that, the RANS equation can also be solved.

2.5.1 The k-ε model

Since the realizable k-ε model and k-ω model will be used in this project, the basic equations for both models will be described in detail:

The k-ε model is the most commonly used of all the turbulence models. It is classified as a two equation model. This denotes the fact that the transport equation is solved for two turbulent quantities k and ε. Within the model the properties k and ε are defined through equation below:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (2.4)$$

C_μ represents the dimensionless parameter or a constant, k is the turbulent kinetic energy, and ε, is the turbulent dissipation. The turbulent kinetic energy is modelled by the transport equation given below:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \overline{u_j k})}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial k}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left(\frac{\rho}{2} \overline{u_j u_i u_i} + \overline{p u_j} \right) - \rho \overline{u_i u_j} \frac{\partial \overline{u_i}}{\partial x_k} \frac{\partial \overline{u_i}}{\partial x_k} \quad (2.5)$$

The second term on the left hand side of the equation and the first term on the right can be solved easily. The second term on the right hand side is known as turbulent diffusion and another model is needed to solve this term. The third term on the right hand side represents the rate of production of kinetic energy by the mean flow, P_k , while the last term represents the rate of dissipation of kinetic energy into internal energy, ε. These two terms also needs to be modelled, with ε needing its own transport equation. This is given by the equation below:

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \overline{u_j \varepsilon})}{\partial x_j} = C_{\varepsilon 1} P_k \frac{\varepsilon}{k} - \rho C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left(\frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) \quad (2.6)$$

where $C_{\epsilon 1}$, $C_{\epsilon 2}$, and σ_{ϵ} , are again dimensionless parameters and constants. These dimensionless parameters are set depending the performance of the model solving known simple test cases, and can be changed whenever necessary.

2.5.2 The k- ω model

For the k- ω model, the turbulent viscosity is defined as

$$\mu_t = \rho \frac{k^2}{\omega} \quad (2.7)$$

with the k equation being slightly modified, such that

$$\frac{\partial(\rho k)}{\partial t} + \partial \frac{(\rho \overline{u_j k})}{\partial x_j} = P_k - \rho \beta^* k \omega + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k^*} \right) \frac{\partial \omega}{\partial x_j} \right] \quad (2.8)$$

which contains the production, diffusion and dissipation term as before. The equation given by:

$$\frac{\partial(\rho \omega)}{\partial t} + \partial \frac{(\rho \overline{u_j \omega})}{\partial x_j} = \alpha \frac{\omega}{k} P_k - \rho \beta \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k^*} \right) \frac{\partial \omega}{\partial x_j} \right] \quad (2.9)$$

where $\alpha, \beta, \beta^*, \alpha_k^*, \alpha_m^*$ which can be changed whenever necessary. From practice, it can be seen that each of these models give better results at different flow conditions, thus it is important to choose the best model depending on the flow problem that is needs to be solved.

2.5.3 Near Wall Treatment Condition

An important issue in turbulence modeling is the numerical treatment of the equations in regions close to the walls. When there is a wall, the turbulence becomes more complex because of the no slip condition at the wall where the flow is reduced to laminar flow or the molecular viscosity dominates. The near wall formulation determines the accuracy of the wall shear stress and heat transfer predictions.

The walls affect the turbulent flow by:

1. No-slip condition at the wall which is $v(\text{wall}) = 0$.
2. Viscous damping and kinematics blocking near the wall reduce velocity fluctuations.
3. Large gradients in temperature and velocity field occur near the wall because walls are the main source of turbulence.
4. The near wall treatment determines how the solver analyses this near wall region. In general there are 3 near wall treatments available, using
 - standard wall functions
 - non-linear wall functions

A wall function is a distribution function that describes the variation of U_i , T , $u_i u_j$, k , and ϵ between a wall and the turbulent zone near it. It is often used to bypass the necessity of detailed numerical treatment and the uncertain validity of a turbulence model.

2.5.4 Discretization Method

Discretization involves the substitution of a variety of finite-difference-type approximations for the terms in the integrated equations representing flow processes such as convection, diffusion and sources. This converts the integral